

## Example 2g: User-Defined Material Constitutive Model

---

This example problem illustrates how to employ the user-defined subroutine, `usrmat.F90`, for implementation of a user's own constitutive model. The appropriate code that calculates the inelastic strain increments for the user's constitutive model must be placed in the file `usrmat.F90`. As was the case in the previous example, this subroutine must be compiled and linked with the MAC/GMC 4.0 library file in order to be used. In this example, a power law creep model and the Bodner-Partom model have been coded within `usrmat.F90`. An additional user-defined subroutine must also be implemented in conjunction with the `usrmat.F90` routine. This subroutine, called `usrformde.F90`, is employed to calculate the stiffness matrix for the user-defined material. In general, since a user-defined constitutive model can employ any elastic material parameters desired by the user, the code will not know how to determine the required stiffness matrix from the user's elastic material parameters. Thus, the user must provide the code to do this in `usrformde.F90`. The user-defined portions of the `usrmat.F90` and `usrformde.F90` subroutines are given in the Appendix. The full listing of these subroutines is provided in the MAC/GMC 4.0 Keywords Manual Appendix.

Note that the specification of the material properties is somewhat different for a user-defined constitutive model. Particularly, the number of elastic and inelastic parameters that code should read in must be specified. The present example also includes several types of mechanical loading. In addition to the standard applied strain history, this problem involves a stress-controlled creep test simulation in which a stress is applied to the material and then held constant as the material creeps.

### MAC/GMC Input File: `example_2g.mac`

```
MAC/GMC 4.0 Example 2g - USRMAT Constitutive Model
*CONSTITUENTS
  NMATS=2
  M=1 CMOD=99 MATID=U MATDB=1 NPE=2 NPV=2 &
    EL=55.2E9,0.30 ALP=22.5E-6,22.5E-6 VI=1.5E-28,3.0
  M=2 CMOD=99 MATID=U MATDB=1 NPE=2 NPV=6 &
    EL=55.2E9,0.30 ALP=22.5E-6,22.5E-6 &
    VI=1000.,103.42E6,103.42E6,1700.,10.,1.0
*RUC
# -- Alter value of M=* to change simulated material
  MOD=1 M=1
*MECH
  LOP=1
  NPT=2 TI=0.,200. MAG=0.,0.02 MODE=1
# NPT=3 TI=0.,10.,3600. MAG=0.,87.2E6,87.2E6 MODE=2,2
# NPT=3 TI=0.,10.,3600. MAG=0.,80.E6,80.E6 MODE=2,2
*THERM
  NPT=2 TI=0.,200. TEMP=23.,23.
# NPT=3 TI=0.,10.,3600. TEMP=23.,23.,23.
*SOLVER
  METHOD=1 NPT=2 TI=0.,200. STP=0.05
# METHOD=1 NPT=3 TI=0.,10.,3600. STP=0.05,0.05
*PRINT
  NPL=6
*XYPLOT
```

```

FREQ=20
MACRO=2
  NAME=example_2g X=1 Y=7
  NAME=example_2g X=101 Y=1
MICRO=0
*END

```

## Annotated Input Data

1) Flags: None

2) Constituent materials (**\*CONSTITUENTS**) [KM\_2]:

```

NMATS=2
M=1 CMOD=99 MATID=U MATDB=1 NPE=2 NPV=2 &
  EL=55.2E9,0.30 ALP=22.5E-6,22.5E-6 VI=1.5E-28,3.0
M=2 CMOD=99 MATID=U MATDB=1 NPE=2 NPV=6 &
  EL=55.2E9,0.30 ALP=22.5E-6,22.5E-6 &
  VI=1000.,103.42E6,103.42E6,1700.,10.,1.0

```

Number of materials:	2	(NMATS=2)
Constitutive models:	User-defined	(CMOD=99)
Materials:	User-defined	(MATID=U)
Material property source:	Input file	(MATDB=1)
No. of Elastic Props:	2	(NPE=2)
No. of Viscoplastic Props:	2, 6	(NPV=2, NPV=6)

Since the materials employ user-defined constitutive models, the number of elastic (NPE) and viscoplastic (NPV) material parameters must be specified by the user. Then, the elastic and viscoplastic material parameters are listed as EL=... and VI=..., respectively. Temperature-dependent material parameters may be specified as well, for details, see the MAC/GMC 4.0 Keywords Manual Section 2.

3) Analysis type (**\*RUC**) → Repeating Unit Cell Analysis [KM\_3]:

Analysis model:	Monolithic material	(MOD=1)
Material assignment:	Each constituent successively	(M=*)

4) Loading:

a) Mechanical (**\*MECH**) [KM\_4]:

```

LOP=1
NPT=2 TI=0.,200. MAG=0.,0.02 MODE=1
# NPT=3 TI=0.,10.,3600. MAG=0.,87.2E6,87.2E6 MODE=2,2
# NPT=3 TI=0.,10.,3600. MAG=0.,80.E6,80.E6 MODE=2,2

```

Loading option:	1	(LOP=1)
Number of points:	2 or 3	(NPT=2, NPT=3)
Time points:	0., 200. sec.	(TI=0., 200.)
	0., 10., 3600. sec.	(TI=0., 10., 3600.)
Load magnitude:	0., 0.02	(MAG=0., 0.02)
	0., 80., 80. MPa	(MAG=0., 80.E6, 80.E6)
	0., 87.2, 87.2 MPa	(MAG=0., 87.2E6, 87.2E6)

Loading mode:	strain control	(MODE=1)
	stress control	(MODE=2 , 2)

Two types of mechanical loading are specified that can be switched by commenting and uncommenting the appropriate lines. The first line specifies mechanical loading in the form of a standard strain-controlled (MODE=1) stress-strain test. The second and third lines specify simulated stress-controlled (MODE=2) creep tests at two different stress levels. In these simulated creep tests, the load is applied quickly in stress control, and then held for 3600 sec. During this hold time, the material is free to creep.

b) Thermal (**\*THERM**) [KM\_4]:

Number of points:	2 or 3	(NPT=2 or NPT=3)
Time points:	0., 200. sec.	(TI=0 . , 200 .)
	0., 10., 3600. sec.	(TI=0 . , 10 . , 3600 .)
Temperature points:	23., 23. °C	(TEMP=23 . , 23 .)
	23., 23., 23. °C	(TEMP=23 . , 23 . , 23 .)

By commenting and uncommenting the appropriate lines, the thermal time points should match those in the mechanical loading under **\*MECH**.

c) Time integration (**\*SOLVER**) [KM\_4]:

Time integration method:	Forward Euler	(METHOD=1)
Number of points:	2 or 3	(NPT=2 or NPT=3)
Time points:	0., 200. sec.	(TI=0 . , 200 .)
	0., 10., 3600. sec.	(TI=0 . , 10 . , 3600 .)
Time step sizes:	0.05 sec.	(STP=0 .05)
	0.05, 0.05 sec.	(STP=0 .05 , 0 .05)

Again, the appropriate lines should be commented and uncommented such that the appropriate time profile is specified for the desired case.

5) Damage and Failure: None

6) Output:

a) Output file print level (**\*PRINT**) [KM\_6]:

Print level:	6	(NPL=6)
--------------	---	---------

b) x-y plots (**\*XYPLOT**) [KM\_6]:

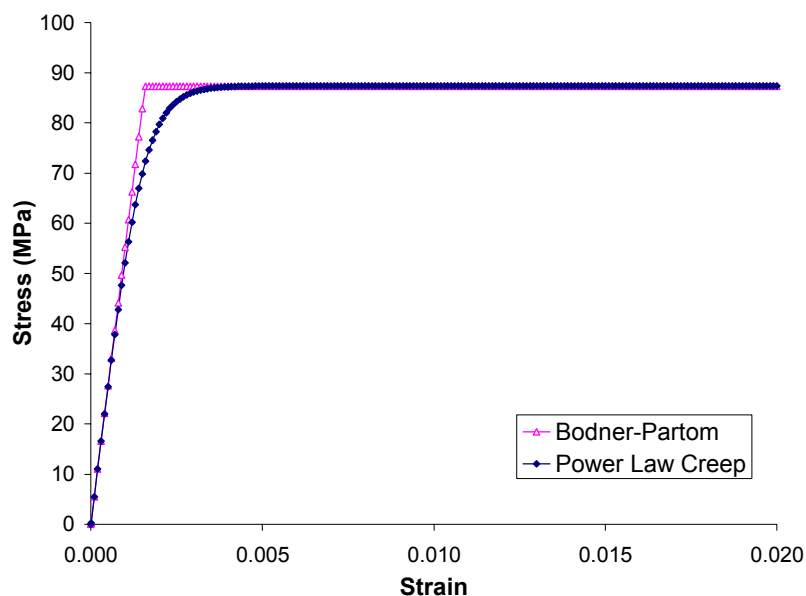
Frequency:	20	(FREQ=20)
Number of macro plots:	2	(MACRO=2)
Macro plot names:	example_2g	(NAME=example_2g)
	example_2g	(NAME=example_2g)
Macro plot x-y quantities:	$\epsilon_{11}$ , $\sigma_{11}$	(X=1 Y=7)
	time, $\epsilon_{11}$	(X=101 Y=7)
Number of micro plots:	0	(MICRO=0)

Since both macro x-y plot files are given the same name, the number 2 will be appended to the name of the second file. Thus, this input file will cause the code to write x-y plot data to the files: `example_2g_macro.data` and `example_2g2_macro.data`. See the MAC/GMC 4.0 Keywords Manuals Section 6 for details.

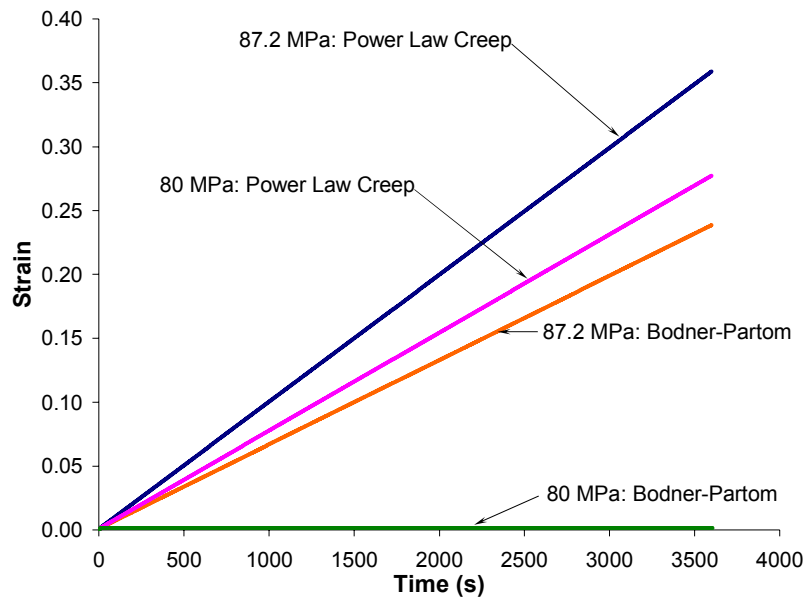
7) End of file keyword: (**\*END**)

## Results

Figure 2.10 shows that at an applied strain rate of  $10^{-4}$ /sec., the user-defined power law creep and Bodner-Partom materials exhibit similar tensile stress-strain behavior. However, as shown in Figure 2.11, the creep behavior of the two materials is completely different at both applied stress levels. Note that because this example problem employs the versions of the user-defined subroutines (`usrmat.F90` and `usrformde.F90`) that are distributed with MAC/GMC 4.0, this example problem cannot be expected to yield the same results if the code is calling altered version of these subroutines. Conversely, the user should ensure that the distributed versions of these subroutines are not being called when the intention is to have MAC/GMC 4.0 call the user's own versions of these subroutines.



**Figure 2.10** Example 2g: plot of the tensile stress-strain response for user-defined material constitutive models.



**Figure 2.11** Example 2g: plot of the creep response for user-defined material constitutive models at two applied stress levels.